

Autodesk Inventor Fusion Sample Files

Autodesk Inventor 2023 Essentials Plus
 Autodesk Inventor 2021
 Autodesk Fusion 360: Introduction to Surface and T-Spline Modeling
 Autodesk Fusion 360 Black Book (Colored)
 Autodesk Fusion 360 Black Book (V 2.0.6508)
 Mastering Autodesk Inventor 2010
 Tinkercad For Dummies
 3D Printing with Autodesk
 Mastering Autodesk Inventor 2015 and Autodesk Inventor LT 2015
 Mastering Autodesk Inventor 2009 and Autodesk Inventor LT 2009
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 App Inventor 2 Databases and Files
 Autodesk Inventor 2020: Introduction for Experienced 3D CAD Users (Mixed Units) - Part 1
 AUTODESK FUSION 360 BLACK BOOK
 Your First Parametric Designs in Autodesk® Inventor® 2020
 Autodesk Fusion 360 Basics Tutorial
 Autodesk Inventor 2020 Essentials Plus
 Mastering Autodesk Inventor 2016 and Autodesk Inventor LT 2016
 Autodesk Inventor Professional 2021 for Designers, 21st Edition
 Learning Autodesk Inventor 2010
 Autodesk Inventor from the Top
 Autodesk Fusion 360 Black Book (V 2.0.10027)
 Autodesk Inventor 2021 Parametric Design and ILogic for Beginners
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 Engineering Design Graphics with Autodesk Inventor 2017
 Mastering AutoCAD 2013 and AutoCAD LT 2013
 Parametric Modeling with Autodesk Inventor 2020
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 Tools for Design Using Autocad 2014 and Autodesk Inventor 2014
 Parametric Modeling with Autodesk Inventor 2018
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 Autodesk Inventor Fusion Sample Files *guest*

KATELYN MARSHALL

Autodesk Inventor 2023 Essentials Plus SDC Publications
 Using a step-by-step format, Engineering Design Graphics with Autodesk Inventor 2017 shows students how to use Autodesk Inventor to create and document designs. Chapter test questions help students assess their understanding of key concepts. Sample problems, end-of-chapter projects, and a variety of additional exercises reinforce the material and allow students to practice the techniques described. The content of the book goes beyond the material normally presented in an engineering graphics text associated with CAD software to include exercises requiring students to design simple mechanisms. This book includes the following features: Step-by-step format throughout the text allows students to work directly from the text to the screen and provides an excellent reference during and after the course. Exercises, sample problems and projects appear in each chapter, providing examples of software capabilities and giving students an opportunity to apply their own knowledge to realistic design situations. Includes examples of how to create an animated assembly, apply dimension to a drawing, calculate shear and bending values, and more! ANSI and ISO standards are discussed when appropriate, introducing students to both so they learn appropriate techniques and national standards.

Autodesk Inventor 2021 John Wiley & Sons
 Autodesk Fusion is a product of Autodesk Inc. It is the first of its kind of software which combine D CAD, CAM, and CAE tool in single package. It connects your entire product development process in a single cloud based platform that works on both Mac and PC. In CAD environment, you can create the model with parametric designing and dimensioning. The CAD environment is equally applicable for assembly design. The CAE environment facilitates to analysis the model under real-world load conditions. Once the model is as per your requirement then generate the NC program using the CAM environment. With lots of features and thorough review, we present a book to help professionals as well as beginners in creating some of the most complex solid models. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between educational and industrial use of Autodesk Fusion. In this edition of book, we have included topics on Sketching, D Part Designing, Assembly Design, Rendering & Animation, Sculpting, Mesh Design, CAM, Simulation, D printing, D PDFs. Contents Starting with Autodesk Fusion 360 Sketching 3D Sketch and Solid Modelling Advanced 3D Modelling Practical and Practice Solid Editing Assembly

Design Importing Files and Inspection Surface Modelling Rendering and Animation Drawing Sculpting Sculpting-2 Mesh
 Design CAM Generating Milling Toolpaths - 1 Generating Milling Toolpaths - 2 Generating Turning and Cutting
 Toolpaths Miscellaneous CAM Tools Introduction to Simulation in Fusion 360 Simulation Studies in Fusion 360

Autodesk Fusion 360: Introduction to Surface and T-Spline Modeling John Wiley & Sons

Learn Autodesk Inventor 2010 in this full-color Official Training Guide This Official Training Guide from Autodesk is the perfect resource for beginners or professionals seeking training or preparing for certification in Autodesk's Inventor 3D mechanical design software. With instruction provided by experts who helped create the software, the book thoroughly covers Inventor principles and fundamentals, including 3D parametric part and assembly design, digital prototyping, and the creation of production-ready drawings. In eye-popping full color, the book includes pages of screen shots, step-by-step instruction, and real-world examples that both instruct and inspire. Takes you under the hood of Inventor 2010, Autodesk's 3D mechanical design software; this book is an Autodesk Official Training Guide Offers Autodesk's own, proven Inventor techniques, workflows, and content tailored to those developing their skills as well as professionals preparing for Inventor certification Teaches 3D parametric part and assembly design, digital prototyping, annotation, dimensioning, and drawing standards Demonstrates best practices for grouping parts into assemblies-then editing, manipulating, and creating drawings Illustrates in full-color with real-world designs, examples, and screen shots Learn Autodesk Inventor 2010 and prepare for Inventor certification with this in-depth guide.

Autodesk Fusion 360 Black Book (Colored) John Wiley & Sons
 Parametric Modeling with Autodesk Inventor 2018 contains a series of seventeen tutorial style lessons designed to introduce Autodesk Inventor, solid modeling, and parametric modeling. It uses a hands-on, exercise-intensive approach to all the important parametric modeling techniques and concepts. The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, creating multi-view drawings and assembly models. Other featured topics include sheet metal design, motion analysis, 2D design reuse, collision and contact, stress analysis, 3D printing and the Autodesk Inventor 2018 Certified User Examination.

Autodesk Fusion 360 Black Book (V 2.0.6508) CAD Artifex
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parametric modeling techniques and concepts. The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, to creating multi-view drawings and assembly models. Other featured topics include sheet metal design, motion analysis, 2D design reuse, collision and contact, stress analysis, 3D printing and the Autodesk Inventor 2020 Certified User Examination. Autodesk Inventor 2020 Certified User Examination The content of Parametric Modeling with Autodesk Inventor 2020 covers the performance tasks that have been identified by Autodesk as being included on the Autodesk Inventor 2020 Certified User examination. Special reference guides show students where the performance tasks are covered in the book. **Mastering Autodesk Inventor 2010** Autodesk Press
 3D Printing with Autodesk Create and Print 3D Objects with 123D, AutoCAD, and Inventor Create amazing 3D-printable objects fast with Autodesk 123D! Imagine it. Then print it! Autodesk 123D gives you all the tools you need and it's free. This easy, full-color guide will help you fully master 3D printing with Autodesk 123D even if you've never done any of this before. Authors John Biehler and Bill Fane have helped thousands of people join the 3D printing revolution—now it's your turn. With step-by-step photos and simple projects, they teach you how to make the most of the whole 123D suite on Windows, Mac, and iPad. New to 3D printing? You'll learn pro techniques for creating models that print perfectly the first time. Want to start fast? Discover how to scan photos straight into your models. Don't have a 3D printer? Learn how to work with today's most popular 3D printing services. John Biehler discovered 3D printing several years ago and built his first 3D printer shortly thereafter. Since then, he's shared his 3D printing knowledge with thousands of people at live events throughout Canada and the Pacific Northwest and through online and broadcast media. He co-founded Vancouver's fastest-growing group of 3D printing enthusiasts. Bill Fane, an Autodesk Authorized Training Centre (ATC) certified instructor, has designed with AutoCAD since 1986. Fane has lectured on AutoCAD and Inventor at Autodesk University since 1995, and at Destination Desktop since 2003. He has written 220 The Learning Curve AutoCAD tutorials for CADalyst and holds 12 patents. From start to finish, 3D Printing with Autodesk 123D covers all you need to know. So stop waiting and start creating! Quickly get comfortable with the 123D workspace and key features Learn the essentials of effective 3D object design Practice 3D design hands-on with simple guided exercises Generate detailed models from photos with 123D Catch Create new 3D character "monsters" with 123D Creature Prepare any 3D model for successful printing Move from existing 3D CAD tools (if you've ever used them) Design parts that are easy to print, and multi-part models that can be printed "pre-assembled" Print through leading 3D printing

services such as Shapeways, Ponoko, Fablab, and Hackerspaces *Tinkercad For Dummies* John Wiley & Sons Autodesk Inventor Professional 2021 for Designers is a comprehensive book that introduces the users to Autodesk Inventor 2021, a feature-based 3D parametric solid modeling software. All environments of this solid modeling software are covered in this book with a thorough explanation of commands, options, and their applications to create real-world products. The mechanical engineering industry examples that are used as tutorials and the related additional exercises at the end of each chapter help the users to understand the design techniques used in the industry to design a product. Additionally, the author emphasizes on the solid modelling techniques that will improve the productivity and efficiency of the users. After reading this book, the users will be able to create solid parts, sheet metal parts, assemblies, weldments, drawing views with bill of materials, presentation views to animate the assemblies and apply direct modelling techniques to facilitate rapid design prototyping. Also, the users will learn the editing techniques that are essential for making a successful design. Salient Features: A comprehensive book consisting of 19 chapters organized in a pedagogical sequence. A detailed explanation of all concepts, techniques, commands, and tools of Autodesk Inventor Professional 2021. Tutorial approach to explain the concepts. Step-by-step instructions that guide the users through the learning process. Real-world mechanical engineering designs as tutorials and projects. Self-Evaluation Test, Review Questions, and Exercises are given at the end of the chapters Table of Contents Chapter 1: Introduction Chapter 2: Drawing Sketches for Solid Models Chapter 3: Adding Constraints and Dimensions to Sketches Chapter 4: Editing, Extruding, and Revolving the Sketches Chapter 5: Other Sketching and Modeling Options Chapter 6: Advanced Modeling Tools-I Chapter 7: Editing Features and Adding Automatic Dimensions to Sketches Chapter 8: Advanced Modeling Tools-II Chapter 9: Assembly Modeling-I Chapter 10: Assembly Modeling-II Chapter 11: Working with Drawing Views-I Chapter 12: Working with Drawing Views-II Chapter 13: Presentation Module Chapter 14: Working with Sheet Metal Components Chapter 15: Introduction to Stress Analysis Chapter 16: Introduction to Weldments (For free download) Chapter 17: Miscellaneous Tools (For free download) Chapter 18: Working with Special Design Tools (For free download) Chapter 19: Introduction to Plastic Mold Design (For free download) Index [3D Printing with Autodesk](#) SDC Publications

- Teaches you the principles of both engineering graphics and Autodesk Inventor 2024
- Uses step by step tutorials that cover the most common features of Autodesk Inventor
- Includes a chapter on stress analysis
- Prepares you for the Autodesk Inventor Certified User Exam

Autodesk Inventor 2024 and Engineering Graphics: An Integrated Approach will teach you the principles of engineering graphics while instructing you on how to use the powerful 3D modeling capabilities of Autodesk Inventor 2024. Using step-by-step tutorials, this text will teach you how to create and read engineering drawings while becoming proficient at using the most common features of Autodesk Inventor. By the end of the book you will be fully prepared to take and pass the Autodesk Inventor Certified User Exam. This text is intended to be used as a training guide for students and professionals. The chapters in this text proceed in a pedagogical fashion to guide you from constructing basic shapes to making complete sets of engineering drawings. This text takes a hands-on, exercise-intensive approach to all the important concepts of Engineering Graphics, as well as in-depth discussions of parametric feature-based CAD techniques. This textbook contains a series of fifteen chapters, with detailed step-by-step tutorial style lessons, designed to introduce beginning CAD users to the graphic language used in all branches of technical industry. This book does not attempt to cover all of Autodesk Inventor 2024's features, only to provide an introduction to the software. It is intended to help you establish a good basis for exploring and growing in the exciting field of Computer Aided Engineering.

Mastering Autodesk Inventor 2015 and Autodesk Inventor LT 2015 SDC Publications

This book/CD-ROM tutorial features numerous examples that relate directly to real-world product design. After chapters on concepts of computer modeling and the functions of Inventor, tutorial chapters cover solid part modeling, sheet metal modeling, assembly modeling, and assembly presentation, featuring step-by-step instructions as well as objectives, overviews, summaries, and review questions. The CD-ROM contains data files for exercises. Cheng teaches at The Industrial Center at Hong Kong Polytechnic University. Annotation (c)2003 Book News, Inc., Portland, OR (booknews.com).

Mastering Autodesk Inventor 2009 and Autodesk Inventor LT 2009 Cadcamcae Works

For courses in Autodesk Inventor, going beyond the available Inventor manuals and references, this text first teaches Inventor step-by-step using many sample and exercise problems - and then shows how to apply it to design problems. Animation of assemblies is included.

Autodesk Inventor 2024: A Power Guide for Beginners and Intermediate Users AutoDesk Press

The complete tutorial and reference to the world's leading CAD program This thoroughly revised and updated edition teaches AutoCAD using explanations, examples, instructions, and hands-on projects for both AutoCAD and AutoCAD LT. This detailed resource works as both a tutorial and stand-alone reference. It introduces the basics of the interface and drafting tools; explores skills such as using hatches, fields, and tables; details such advanced skills as attributes, dynamic blocks, drawing curves, and using solid fills; explains 3D modeling and imaging; and discusses customization and integration. Covers all the new AutoCAD capabilities Written by George Omura, a popular AutoCAD author Offers an essential resource for those preparing for the AutoCAD certification program Includes a DVD with all the project files necessary for the tutorials, a trial version of AutoCAD, and additional tools and utilities George Omura's engaging writing style makes this reference the perfect reference and tutorial for both novice and experienced CAD users. Note: CD-ROM/DVD and other supplementary materials are not included as part of the e-book file, but are available for download after purchase.

Using Autodesk Inventor 6 Serdar Hakan DÜZGÖREN

Autodesk Inventor 2024: A Power Guide for Beginners and Intermediate Users textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers, interested in learning Autodesk Inventor, to create 3D mechanical designs. This textbook is an excellent guide for new Inventor users and a great teaching aid for classroom training. It consists of 14 chapters and a total of 790 pages covering major environments of Autodesk Inventor such as Sketching environment, Part modeling environment, Assembly environment, Presentation environment, and Drawing environment. The textbook teaches you to use Autodesk Inventor mechanical design software for building parametric 3D solid components and assemblies as well as creating animations and 2D drawings. This textbook not only focuses on the usages of the tools/commands of Autodesk Inventor but also on the concept of design. Every chapter in this textbook contains tutorials that provide users with step-by-step instructions for creating mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives that allow users to experience the user-friendly and powerful technical capabilities of Autodesk Inventor. Table of Contents: Chapter 1. Introduction to Autodesk Inventor Chapter 2. Drawing Sketches with Autodesk Inventor Chapter 3. Editing and Modifying Sketches Chapter 4. Applying Constraints and Dimensions Chapter 5. Creating Base Feature of Solid Models Chapter 6. Creating Work Features Chapter 7. Advanced Modeling - I Chapter 8. Advanced Modeling - II Chapter 9. Patterning and Mirroring Chapter 10. Advanced Modeling - III Chapter 11. Working with Assemblies - I Chapter 12. Working with Assemblies - II Chapter 13. Creating Animation and Exploded Views Chapter 14. Working with Drawings [Parametric Modeling with Autodesk Fusion 360](#) Que Publishing Introduction to Using Inventor's Programming Interface There are several resources provided to help you use Inventor's Application Programming Interface (API). These resources are all part of Inventor's Software Development Kit (SDK). The various elements of the SDK and some additional external resources are described below.

Mastering Autodesk Inventor and Autodesk Inventor LT 2011 John Wiley & Sons

Student, designer, engineer? Start your adventure with Autodesk Inventor This book is intended for people for whom this is the first contact with Autodesk Inventor 2021 software. However, individuals who are familiar with the program will find here useful information about using parametrization techniques for the streamline creation of variants of the product. In this manual, you will find extensive descriptions and detailed illustrations explaining the tools used and the correct workflow techniques. The book presents three examples of the use of the software.

Example No 1. Designing a complete product In the first example, you will learn how to work in Inventor, from scratch. You will create a project of a simple drill vise, on which you will learn the basic operations of modeling and creating drawing documentation. This example emphasises the principles of project management, from a single part through designing parts in the context of the assembly, checking the basic kinematics of the product, and further creating a complete drawing documentation containing item numbers and a parts list, as well as an exploding view of the product, rendered illustration and video for marketing purposes. Then, thanks to the program parameterization and skillful file management, you will quickly create a new version of the drill vise with a complete set of drawing documentation as well as a rendered illustration and video of the new version of the product. Example No 2. Component libraries Most of the products being designed, use components purchased from external suppliers. For this reason, parametric 3D models of purchased components, which can be quickly inserted into the project instead of modeling each time from scratch, offer the greatest possible convenience for the constructor. In addition, component library files should be properly described, so that they are correctly presented in the bill of materials and also it should be placed in the library resources area, which will protect them from accidental editing. The examples presented here will teach you

how to prepare your own parametric libraries of purchased components. Example No 3. The parametric generator of product versions In the third example, you will create a parametric generator for making a simple metal casing that allows you to obtain a model of any size, with or without handles and pre-prepared drawing documentation for each version. The generated version of the casing can be further modified in order to obtain the final appearance. In this example, you will learn the basics of designing sheet metal parts, the use of parameters in parts and in the assembly, and you will learn the basics of programming using iLogic and how to use iLogic parametric version generators. And... No additional files for download are required to complete the designs described - all files will be created from scratch in the exercises in sequence. Most of this manual is also compatible with previous versions of Inventor. The completed Table of Contents of this book and set of illustrations of the examples used in the book you can find on: www.expertbooks.eu.

Engineering Design and Graphics with Autodesk Inventor 10 SDC Publications

KEY BENFIT: Using a step-by-step format, this book introduces Autodesk Inventor 10 and shows how to use Autodesk Inventor to create and document designs. Sample problems and a variety of additional exercise problems reinforce the material and allow the reader to practice the techniques described. The content of the book goes beyond the material normally presented in an engineering graphics book associated with CAD software to include exercises requiring users to design simple mechanisms. For users of CAD that want to learn Autodesk Inventor 10.

App Inventor 2 Databases and Files CADArtifex

Autodesk Inventor 2023 Essentials Plus provides the foundation for a hands-on course that covers basic and advanced Autodesk Inventor features used to create, edit, document, and print parts and assemblies. You learn about part and assembly modeling through real-world exercises. Autodesk Inventor 2023 Essentials Plus demonstrates critical CAD concepts, from basic sketching and modeling through advanced modeling techniques, as it equips you with the skills to master this powerful professional tool. The book walks you through every component of the software, including the user interface, toolbars, dialogue boxes, sketch tools, drawing views, assembly modeling, and more. Its unique modular organization puts key information at your fingertips, while step-by-step tutorials make it an ideal resource for self-learning. Packed with vivid illustrations and practical exercises that emphasize modern-day applications, Autodesk Inventor 2023 Essentials Plus will prepare you for work in the real world. Each chapter is organized into four sections. Objectives, which describe the content and learning objectives; topic coverage, which presents a concise review of the topic; exercises, which present the workflow for a specific command or process through illustrated step-by-step instructions; and finally a checking your skills section, which tests your understanding of the material. Who Should Use this Manual? This manual is designed to be used in instructor-led courses, although you may also find it helpful as a self-paced learning tool. It is recommended that you have a working knowledge of Microsoft® Windows® as well as a working knowledge of mechanical design principles.

Autodesk Inventor 2020: Introduction for Experienced 3D CAD Users (Mixed Units) - Part 1 John Wiley & Sons

The Autodesk(R) Inventor(R) 2021: Working with Imported Geometry guide teaches you how to work with data from other CAD platforms using the Autodesk Inventor software. Using this guide, you will learn the various methods for importing data into Autodesk Inventor and how you can edit both imported solid and surface data. Additionally, you will learn how to index scanned point cloud data, and attach and use it in an Inventor file. The final chapters in this guide discuss how you can use AutoCAD .DWG files in the Autodesk Inventor software. The topics covered in this guide are also covered in ASCENT's Autodesk(R) Inventor(R) 2021: Advanced Part Modeling guide, which includes a broader range of advanced learning topics. Topics Covered Import CAD data into the Autodesk Inventor software. Export CAD data from the Autodesk Inventor software in an available export format. Index a supported point cloud data file, attach, and edit it for use in a file. Use the Edit Base Solid environment to edit solids that have been imported into the Autodesk Inventor software. Create Direct Edit features in a model that move, resize, scale, rotate, and delete existing geometry in both imported and native Autodesk Inventor files. Set the import options to import surface data from other file format types. Transfer imported surface data into the Repair Environment to conduct a quality check for errors. Appropriately set the stitch tolerance value so that gaps in the imported geometry can be automatically stitched and identify the gaps that are not stitched. Use the Repair Environment commands to repair gaps or delete, extend, replace, trim and break surfaces to successfully create a solid from the imported geometry. Open an AutoCAD DWG file directly into an Autodesk Inventor part file and review the data. Use the DWG/DXF File Wizard and its options to import files into an Autodesk Inventor file. Use an AutoCAD DWG file in an Autodesk Inventor part file so that the geometry created in Inventor remains associative with the AutoCAD DWG file. Prerequisites Access to the 2021.0 version

of the software, to ensure compatibility with this guide. Future software updates that are released by Autodesk may include changes that are not reflected in this guide. The practices and files included with this guide might not be compatible with prior versions (i.e., 2010). The material covered in this guide assumes a mastery of Autodesk Inventor basics as taught in the Autodesk Inventor: Introduction to Solid Modeling guide.

AUTODESK FUSION 360 BLACK BOOK John Wiley & Sons
 App Inventor 2: Databases and Files is a step-by-step guide to writing apps that use TinyDB, TinyWebDB, Fusion Tables and data files for information storage and retrieval. Includes detailed explanations, examples, and a link to download sample code. This is the first tutorial to cover all of these App Inventor database and file features. If your apps need to work with data or files - you need this book! TinyDB stores data on your smart phone or tablet and is a primary way for App Inventor apps to save data, even when the app is no longer running or if the device is turned off. TinyWebDB is similar to TinyDB, but stores your data on a remote server in the network cloud. Multiple apps can share a TinyWebDB database, plus you can update the content of your TinyWebDB using just a web browser. This means you can distribute an app whose content can change over time - just by changing the values in TinyWebDB. A big challenge is the need to set up a TinyWebDB server - this book shows how to do that through free services offered by Google. Fusion Tables provide a powerful, cloud-based database system for App Inventor apps. Creating, retrieving, updating and deleting data is done using the industry standard Structured Query Language or SQL. Fusion Tables reside in the Google network cloud - this book shows you how to set up and configure Fusion Tables for you own apps using free services of Google. As your app requirements grow, Google's cloud can provide low cost servers and bandwidth for your needs. Underneath the Android OS user interface, there is a file system, similar to the file system found on Windows or Mac OS X. With App Inventor your apps can write and read data from files, and if

using the special "CSV" format, App Inventor data can be shared with many spreadsheet programs. This book shows you how to create, use and access data files, and how to convert data to and from the CSV format. Over 28,000 words. Over 250 screen shots and illustrations. Numerous sample programs and code. App Inventor 2: Databases and Files - Table of Contents 1 - Introduction 2 - Using the TinyDB database 3 - Implementing Records Using Lists in TinyDB 4 - Simulating Multiple TinyDB Databases 5 - How to Use Multiple Tags in TinyDB 6 - Introduction and Setup: TinyWebDB 7 - Managing TinyWebDB in the Cloud 8 - Programming for TinyWebDB - Demo 1 9 - Adding a Tags List to TinyWebDB - Demo 2 10 - Handling Multiple Users with TinyWebDB - Demo 3 11 - Implementing a Student Quiz Application using TinyWebDB 12 - Introduction to Fusion Tables 13 - Developing Your Fusion Table App 14 - Using Text Files in App Inventor
Your First Parametric Designs in Autodesk® Inventor® 2020 SDC Publications
 Note: This book is continued in Autodesk(R) Inventor(R) 2020: Introduction for Experienced 3D CAD Users - Part 2. Both books are required to complete this guide. The Autodesk(R) Inventor(R) 2020: Introduction for Experienced 3D CAD Users learning guide is intended to provide accelerated introductory training in the Autodesk(R) Inventor(R) software. This learning guide is designed for users that have 3D modeling design experience with other 3D CAD software packages (e.g., CATIA(TM), Pro/ENGINEER(R), Creo Parametric(TM), NX(TM), SolidWorks(R), etc.). By leveraging the experience users gain in working with other 3D modeling software packages, this hands-on, practice-intensive guide is developed so that new users in the Autodesk Inventor software can benefit from a shorter, introductory-level, learning guide. You are taught how to find and use the modeling tools associated with familiar modeling strategies that are used in other 3D CAD software. You will acquire the knowledge required to complete the process of creating models from conceptual sketching, through to solid modeling, assembly design, and drawing production. Topics

Covered The Autodesk Inventor software interface Obtaining model information Creating sketch and pick and place features Work Features Creating equations and working with parameters Model geometry and model display manipulation Feature duplication techniques Placing and constraining parts in assemblies Assembly component display Presentation files (Exploded views and Animations) Assembly tools Creating parts and features in assemblies Creating and editing assembly Bill of Materials Working with projects Creating and annotating drawings and views Prerequisites Access to the 2020.0 version of the software, to ensure compatibility with this guide. Future software updates that are released by Autodesk may include changes that are not reflected in this guide. The practices and files included with this guide are not compatible with prior versions (i.e., 2019). Prior knowledge of 3D modeling and 3D CAD software. Users with AutoCAD(R) or AutoCAD(R) Mechanical experience are recommended to use the Autodesk Inventor 2020: Introduction to Solid Modeling guide.
Autodesk Fusion 360 Basics Tutorial Prentice Hall
 Autodesk Fusion is a product of Autodesk Inc. It is the first of its kind of software which combines 3D CAD, CAM, and CAE tool in a single package. It connects your entire product development process in a single cloud-based platform that works on both Mac and PC. In CAD environment, you can create the model with parametric designing and dimension. The CAD environment is equally applicable for assembly design. The CAE environment facilitates to analysis the model under real-world load conditions. Once the model is as per your requirement then generate the NC program using the CAM environment. With lots of features and thorough review, we present a book to help professionals as well as beginners in creating some of the most complex solid models. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between the educational and industrial use of Autodesk Fusion.